5379

Visualising the flow

Computational fluid dynamics is not a new concept but until now has not been tailored to the building services industry. *Mark Seymour* outlines FloVENT, cfd software which brings detailed environment analysis to the design engineer.

For many years, full and part-scale models have been used by designers to assess ventilation performance in building designs, both in terms of the air distribution flow patterns and the resulting velocity and temperature effect on comfort in the occupied zone. While this method is a positive way of determining the likely performance, it is prohibitively expensive for many buildings.

A new design tool, FloVENT, which uses computational fluid dynamics (cfd) has now been developed in an easy to use, industry specific form to bring detailed environment analysis to the engineer. FloVENT has been conceived with the explicit intention of providing cfd software in a form that can be used effectively by non-dedicated users as a method of design assessment for a wider range of projects. The program will predict, where appropriate, velocity, temperature, pressure, and concentration of contaminants. The information can be viewed graphically with diagrams showing flow patterns and temperature distributions in two or three dimensional views.

In recent years, cfd programs have been used successfully at BSRIA and other specialist establishments to assess the performance of ventilation systems. These are general purpose numerical techniques which predict fluid motion, taking into account boundary conditions representing sources, or losses, of mass, heat, pressure etc.

However, the techniques have been of limited scope because of the general-purpose nature of these programs and lack of tailoring to the building services industry. This results in computer modelling failing to achieve the full cost savings compared with physical modelling.

The specification, design and development of this new program has been carried out in a collaboration between BSRIA and Flomerics, making full use of already well developed techniques. The major difference between FloVENT and its general purpose counterparts is that the program has been designed specifically for building services.

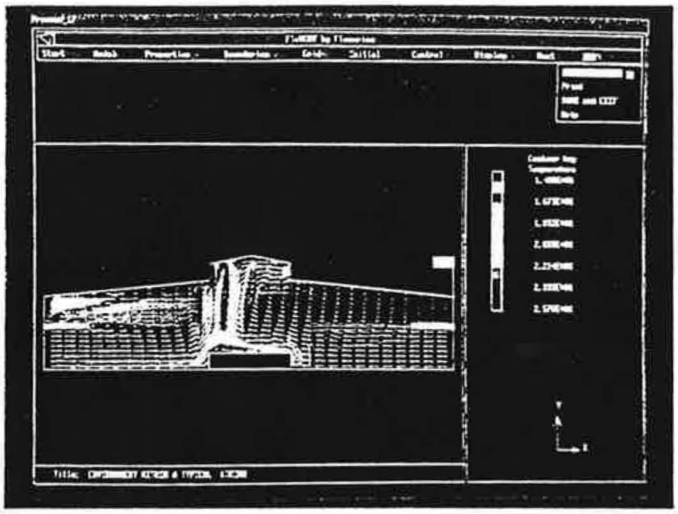

Figure 1: Predicted internal environment within a modern atrium building.

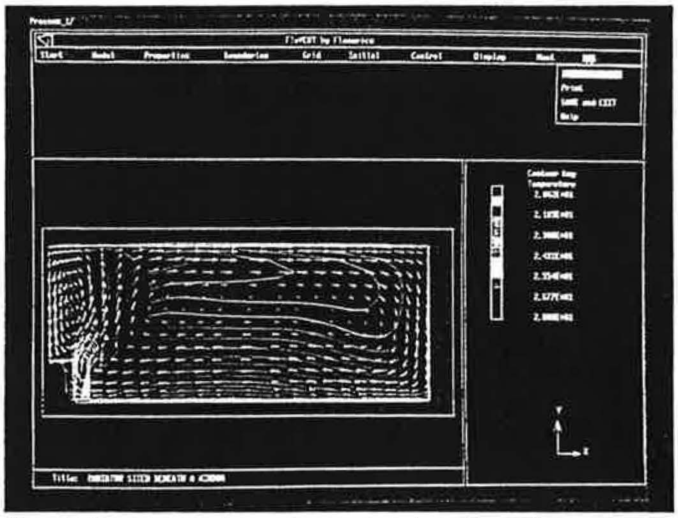

Figure 2: Natural air circulation when a radiator is sited beneath a window.

The ventilation model is constructed using a menu-driven system which requires architectural and hvac information in a form familiar to the professional design engineer.

FloVENT will provide technical features enabling prediciton of the environment both for comfort and equipment. Features include:

☐ 3-dimensional prediction of air movement and temperature with allowance for both natural and forced convection, and including the effects of buoyancy and air turbulence;

☐ menu selection of hvac components (supply, extract, etc) rather than general mass flow boundary conditions;

☐ architectural and internal features of the building, equipment and furniture can be defined by geometrical extent independent of the computational grid, with a built-in materials library available to define their properties, and to which the user can add his own data;

☐ conjugate heat transfer calculations can be carried out to determine the temperature distribution in solid regions as well as the air flow patterns adjacent. This is useful in assessing the likelihood of condensation or where heat capacity of the fabric affects performance during periods such as building heat up;

☐ in addition to air temperature, mean radiant temperature and dry resultant temperature can be calculated so that comfort can be fully assessed;

☐ air turbulence – the "laminar" air viscosity is replaced by an enhanced "effective turbulent viscosity" to account for the effects of air turbulence;

☐ buoyancy – air density is calculated as a function of temperature, and the buoyancy forces thus derived are included in the momentum conservation equation in the vertical direction. The buoyancy forces can give rise to important thermal stratification effects which are predicted by FloVENT.

☐ full on-line help menu describing options and parameters;

☐ the program will be available on PC 386 computers upwards, with interactive graphics available throughout.

In addition to forced ventilation, the program can be used to predict the environment resulting from natural ventilation. Figure 1 shows the calculated environment in a typical modern atrium, with a solar angle of 60°C providing asymmetric radiative gains, with ventilation air being drawn in at 15°C and exiting through roof ventilators. Clearly these techniques provide the opportunity of assessing different ventilation strategies for building types where previously there has been great uncertainty.

Similarly, the air distribution may be a result of heat emission rather than ventilation. Figure 2 shows a strong convective plume from a radiator interacting with the cold downdraught from a window above. In each case the arrows demonstrate the magnitude and direction of air velocity while the coloured contours are isotherms with temperature defined by the adjacent bar.

The approach – the power of cfd techniques to simulate air flow combined with an easy to use menu system for problem definition, solution and result analysis – provides the flexibility to easily assess different ventilation schemes and thus make educated design decisions.

Mark Seymour is head of the Air Distribution Section at the Building Services Research and Information Association in Bracknell.